

# Autodesk Manufacturing & Digital Prototyping Solutions

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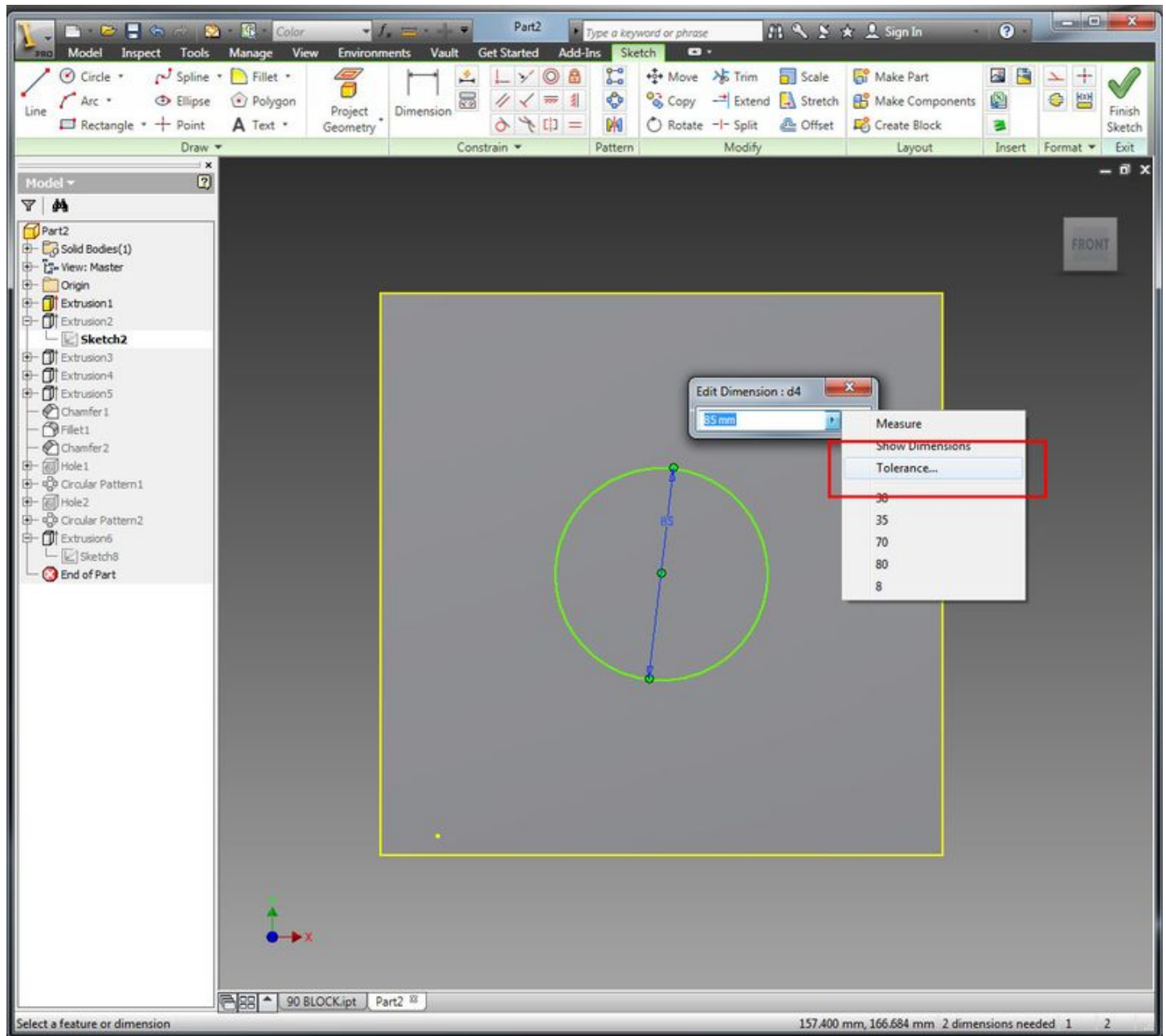
## Autodesk Inventor: Quick Tip - Tolerances

When manufacturing parts it is common practice to add tolerances to dimensions to aid manufacturing. In some cases an overall or general tolerance is shown the drawing being used for manufacture, and in other cases individual tolerances are applied to dimensions on the drawing.

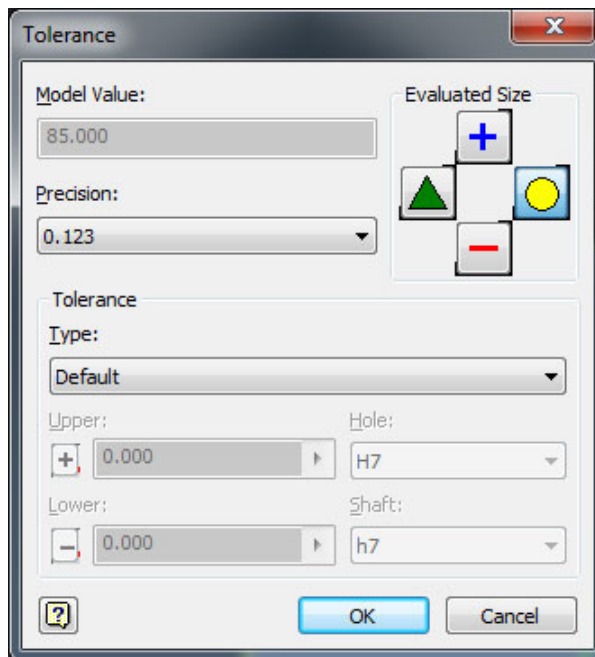
Autodesk Inventor allows users to apply tolerance values to dimensions in the 3D part model which can be then be used to automatically annotate the associative 2D drawing but can also be used to show the 3D model at the different tolerance sizes to check for fit and function.

First, we're going to have a look at adding tolerance values to our 3D part model and using them in the 2D drawing for manufacturing.

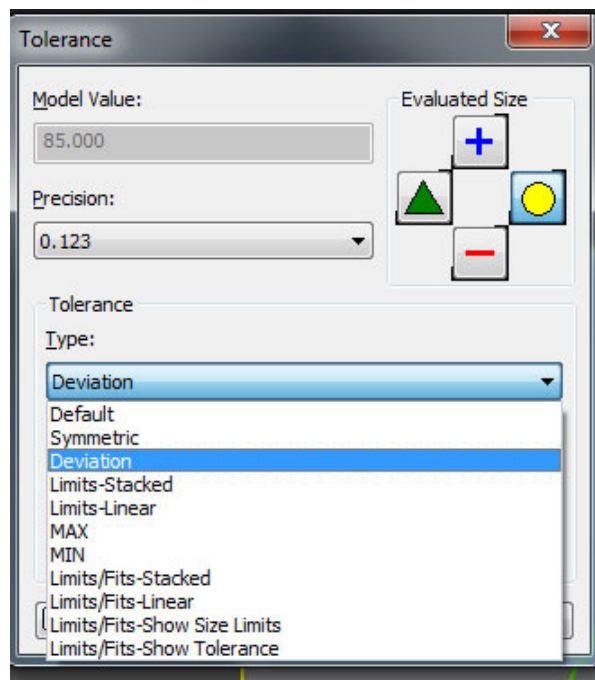
**Step 1:** Adding tolerance values is really easy, just click on the > arrow on the end of the Edit Dimension box and select Tolerance from the list...



**Step 2:** The Tolerance dialogue allows you to select the type of tolerance you wish to add, as well as well as set the Precision and set the Evaluated Size (we'll talk more about that another time)...



**Step 3:** Select from the Tolerance Type list, the type of tolerance you would like to apply...



I found the definitions for each of the tolerance types on the [Autodesk Wiki](#)....

#### **Default**

Used if you do not specify a tolerance type.

#### **Symmetric**

Specifies the same value at the upper and lower tolerance range.

#### **Deviation**

Specifies a different value at the upper and lower tolerance range.

#### **Limits - Stacked**

Displays the maximum and minimum tolerance values in a stack.

#### **Limits - Linear**

Displays minimum values and maximum values side by side separated by a hyphen.

#### **MAX**

Displayed as a label on the maximum tolerance value. The upper tolerance is zero and is disabled; the lower tolerance level is any number equal to or greater than zero.

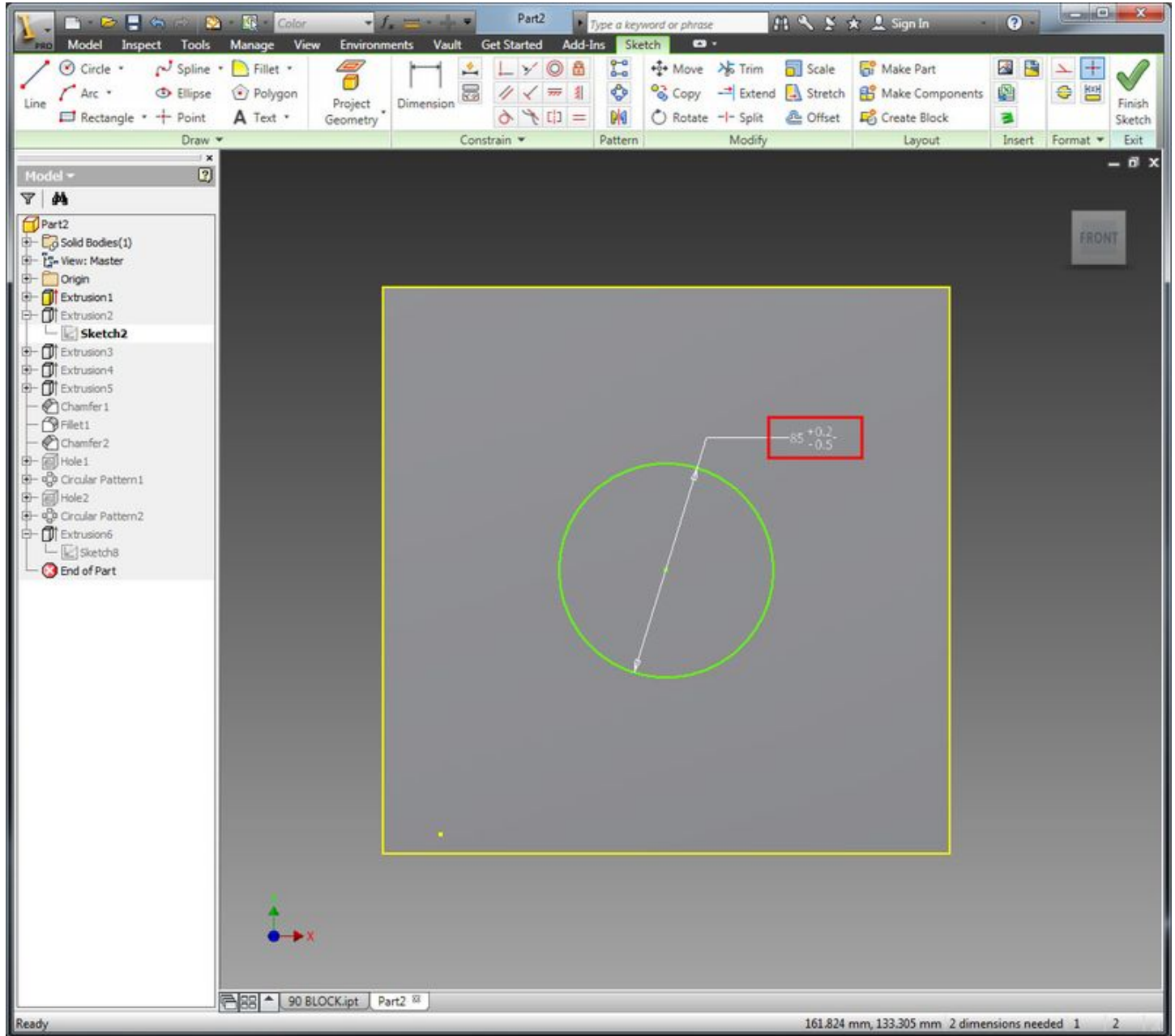
**MIN**

Displayed as a label on the minimum tolerance value. The lower tolerance is zero and is disabled. The upper tolerance is zero and is disabled; the upper tolerance level is any number equal to or greater than zero.

**Limits and Fits**

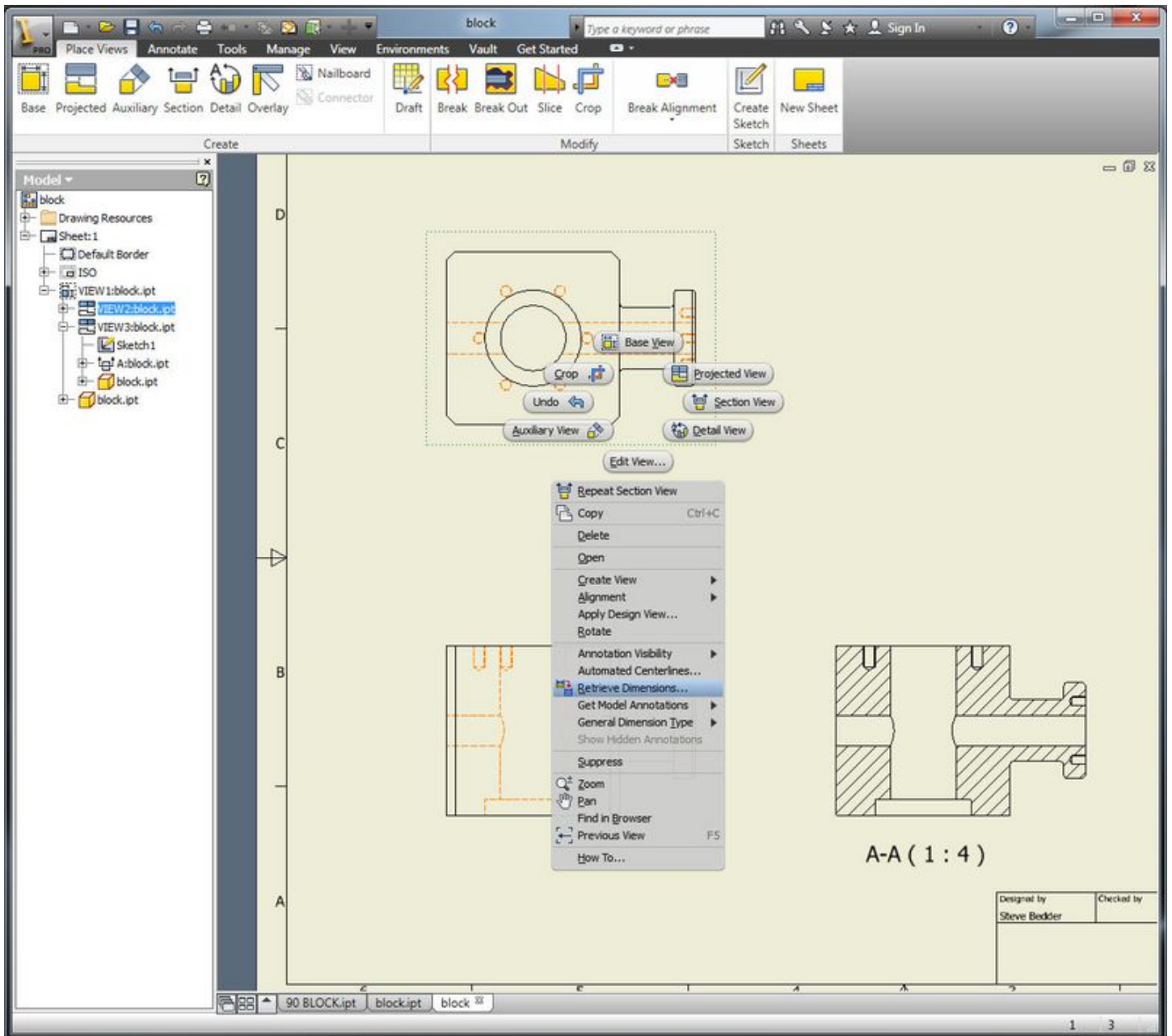
Used to show tolerances on shafts and holes in addition to tolerances for the part. Nomenclature and tolerances are specified by the standard selected (such as ANSI or ISO) when Autodesk Inventor was installed. Limits are the tolerances for a part; fits are the pair of limits for a pair of mated parts. Stacked and Linear options are available. The Show Size Limits option displays the tolerance limits in parentheses. The Show Tolerances option displays values in parentheses with a plus and minus to indicate the upper and lower tolerances.

**Step 5:** Once you've applied the tolerance value it will be shown on the dimension in your 2D sketch...

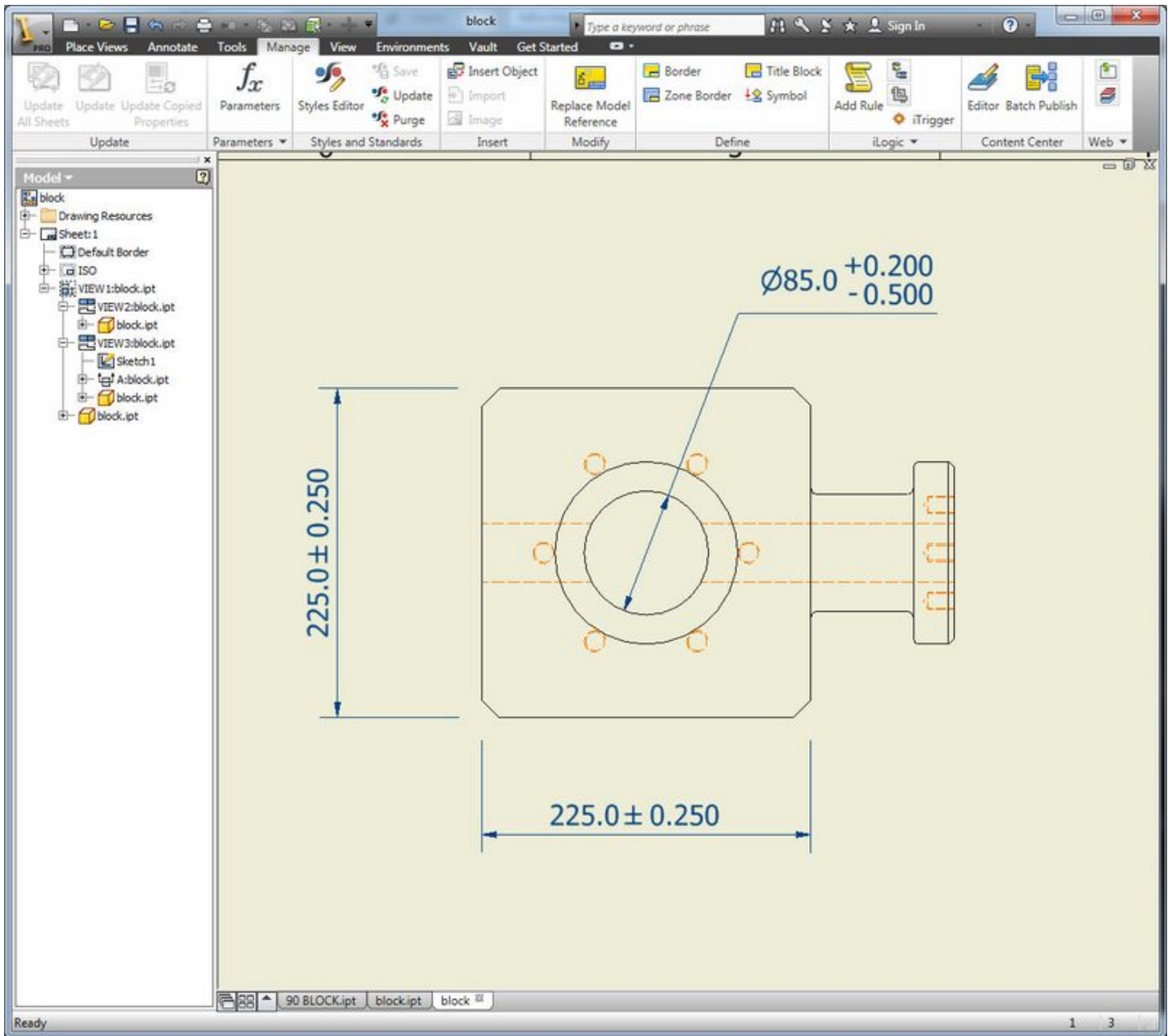


**NB:** If you can't see the tolerance value like it shown above it is most likely due to the Dimension Display settings. Just right mouse click in the sketch and select Dimension Display from the list and make sure there is a tick next to where it says Tolerance.

**Step 6:** Create a 2D drawing view of the part as a DWG or an IDW, right mouse click over the view and select Retrieve Dimensions...



**Step 7:** Select the model dimensions you wish to show on the drawing view from the preview and when they are applied to the view you will see that the tolerance values you applied in the 3D model are automatically annotated to the drawings on the 2D drawing...



The tolerances are associative to the data in the 3D model meaning a change to the values in the 3D part will automatically update on the 2D drawing.

Next time, we'll have a look at how we can use the tolerance tool to automatically re-model that part based on the tolerance values.

Posted by [Steve Bedder](#) on 06/07/2011 in [Inventor](#), [Tips & Tricks](#) | [Permalink](#)

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